CFD Workbench

FreeCAD-CFD Workbench

Tutorial 4: External aerodynamics of a UAV

our future through science

CFD Workbench

WORKBENCH

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

INSTALLATION

WINDOWS:

- https://www.freecadweb.org/wiki/Download
- Install CfdOF from Tools | Addon manager
- Go to Edit | Preferences | CFD to check and install dependencies

LINUX:

- *https://www.freecadweb.org/wiki/Install_on_Unix*
- Install CfdOF from Tools | Addon manager
- Install OpenFOAM (*https://openfoam.com/download/*)
- Install Paraview
- Go to Edit | Preferences | CFD to check dependencies and install cfMesh

LATEST INFORMATION

Please see the CfdOF README file for up-to-date information.

LEAD DEVELOPERS

Johan Heyns (CSIR, 2016-2018) jaheyns@gmail.com,
Oliver Oxtoby (CSIR, 2016-2018) oliveroxtoby@gmail.com,
Alfred Bogaers (CSIR, 2016-2018) alfredbogaers@gmail.com,

UAV aerodynamics

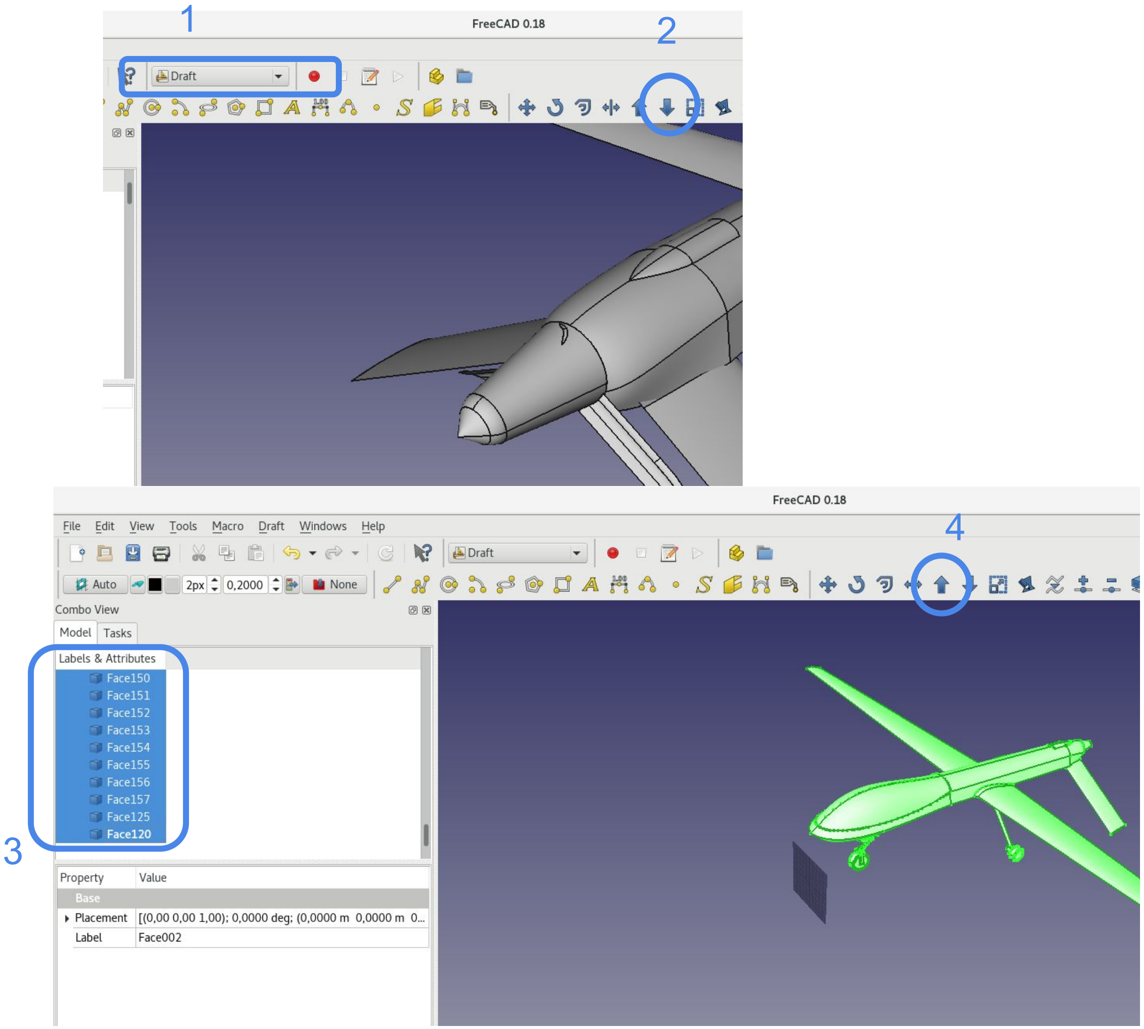
- To demonstrate how to model viscous flow over an unmanned aerial vehicle.
- Study the effect of including the camera gimbal

Created by Shadman Sakib

Part Design

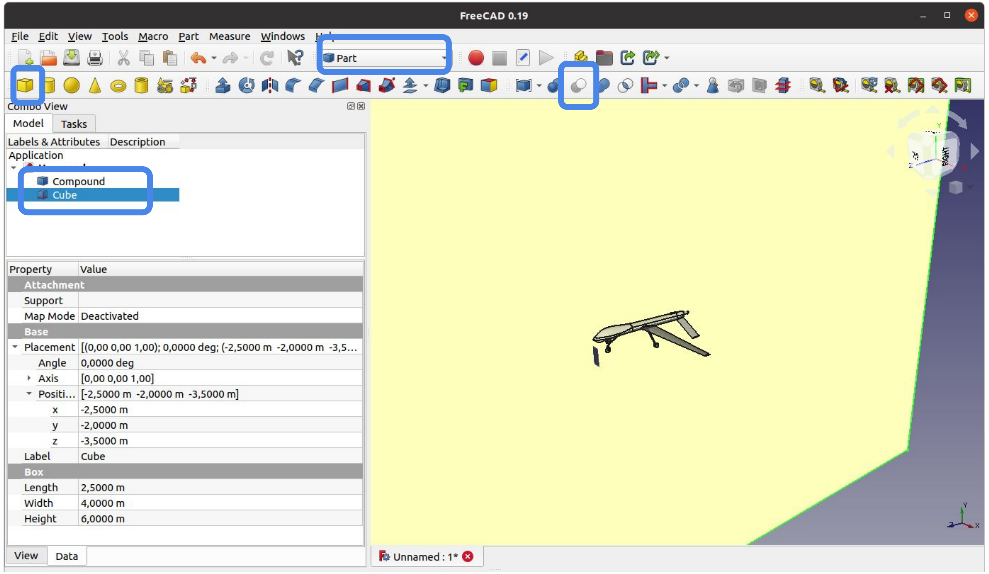
Geometry

- Open the supplied .igs file in FreeCAD
- We wish to remove the propeller blades for the analysis.
- Open the ‘Draft’ workbench, select the ‘UAV’ object in the tree view, and click the ‘Explode’ button.
- Select and delete each face of the propeller.
- Select all faces and click the ‘join’ button to re-combine them.



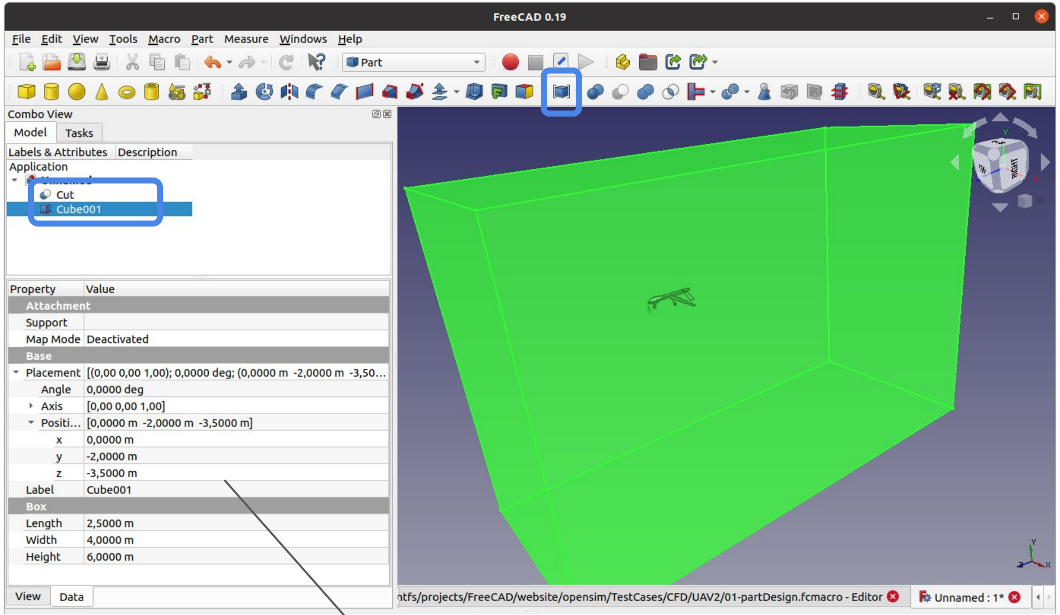
Geometry

- Save time by simulating only the x > 0 half of the symmetric domain - therefore, we wish to cut away half the geometry
- Open the ‘Part’ workbench and create a cube
- Set its dimensions to surround the geometry for x ⩽ 0
- To cut the aircraft geometry with the cube, **first** select the ‘Compound’ object, then the ‘Cube’ object (holding Ctrl), and choose the ‘Cut’ operation in the Part workbench
    - The ‘Cut’ operation cuts away from the first selection using the second (and any further) selections

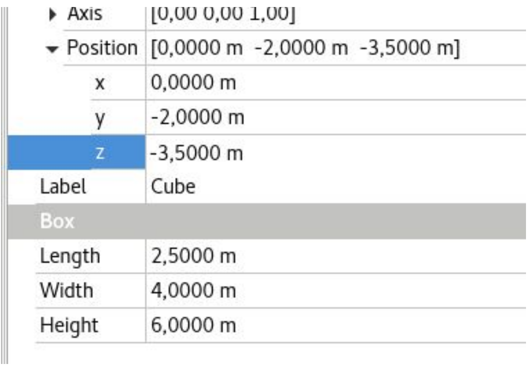


External flow domain

- Create another cube to hold the external mesh.
- Set the position and dimensions as shown.
- For the final analysis, the far field domains should be much further from the body
  - Rule of thumb is 10 times its characteristic length
  - We choose closer boundaries for a quicker preliminary analysis
- Join the surfaces together by selecting them in the tree view and choosing “Make compound”



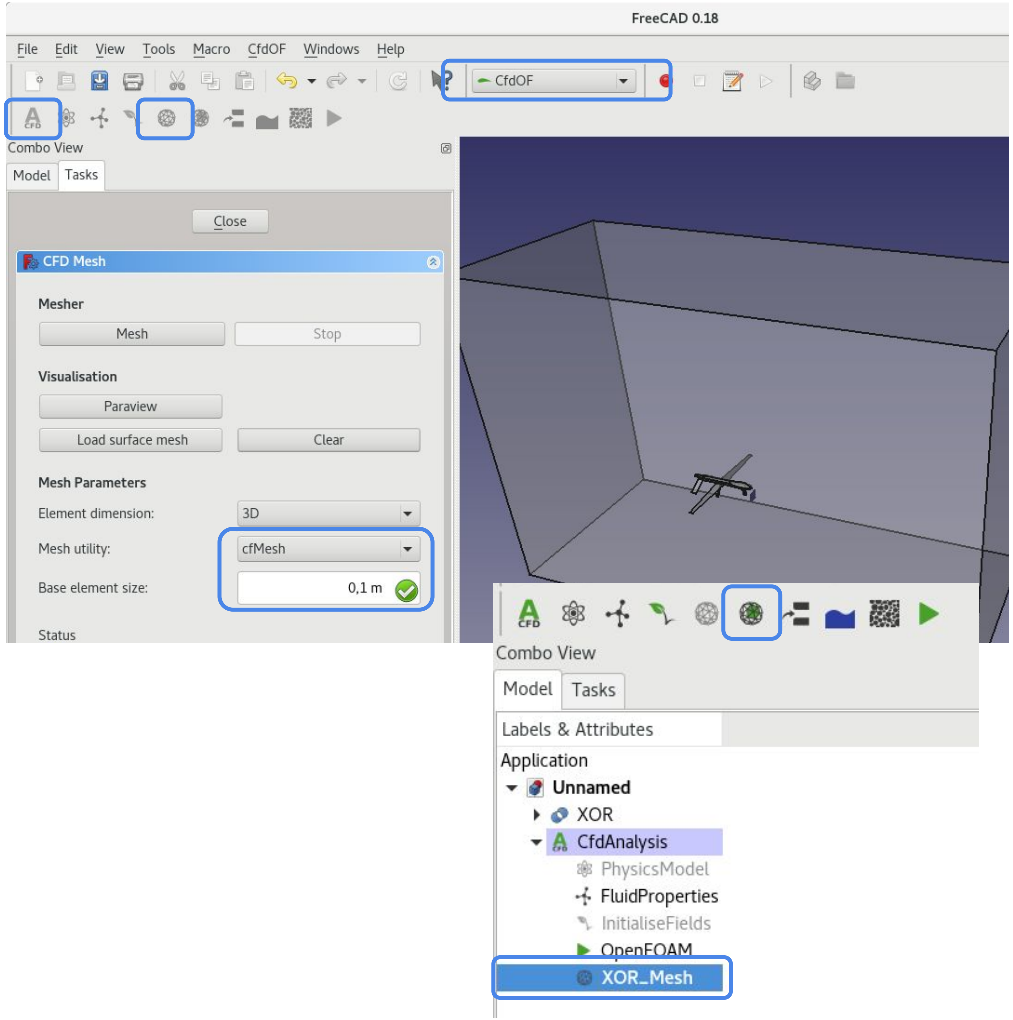
NOTE: Choosing operations that act on surfaces rather than solids allows us to work with non-watertight CAD such as in this example



Mesh generation and mesh refinement with cfMesh

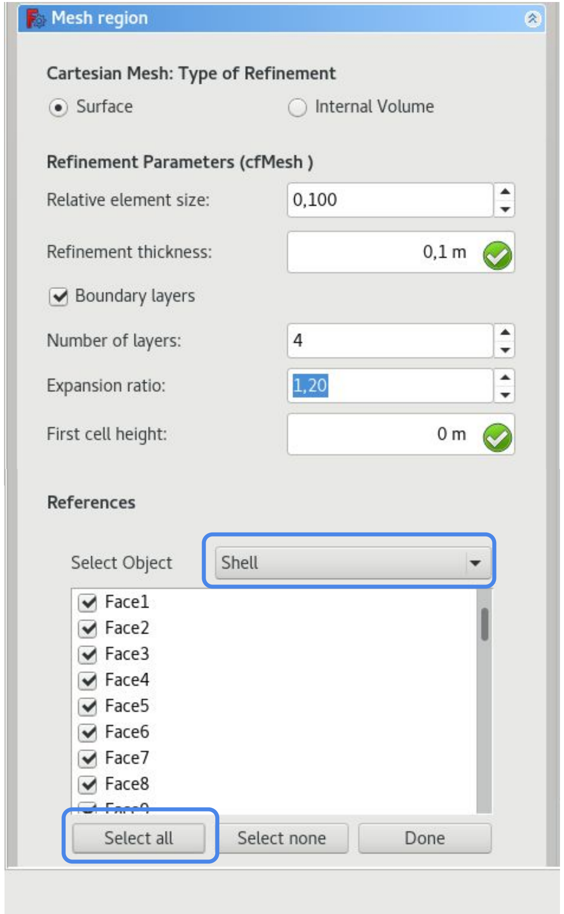
Create Mesh object and refinement region

- Activate the CfdOF Workbench
- Create an ‘Analysis’ object
- Select the ‘Compound001’ object and click the ‘Mesh’ button.
- Select the cfMesh mesher and a base element size of 0.1 m
- Select the mesh object and click the ‘Mesh region’ button



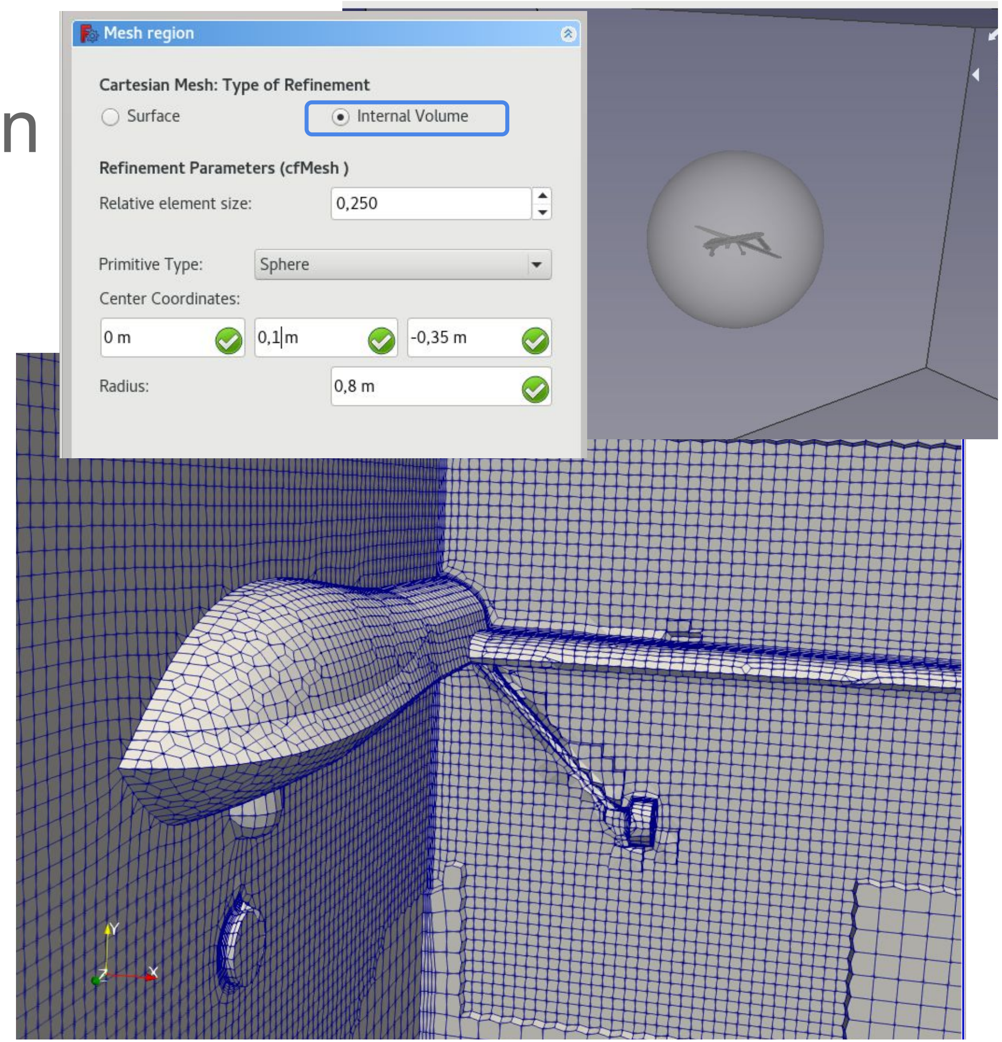
Surface refinement region

- For 1/10th refinement within 0.1 m of the body, input a refinement thickness of 0.1m and a relative element size of 0.1.
- Here we choose 4 boundary layers with an expansion ratio of 1.2.
    - See Tutorial 3 for more information on boundary layers
    - For non-smooth geometries, the mesher and/or solver may struggle if too many boundary layers are added.
- To easily select the entire body of the aircraft, click ‘Select from list’, then choose the ‘Cut’ object and click ‘Select all’



Volume refinement region

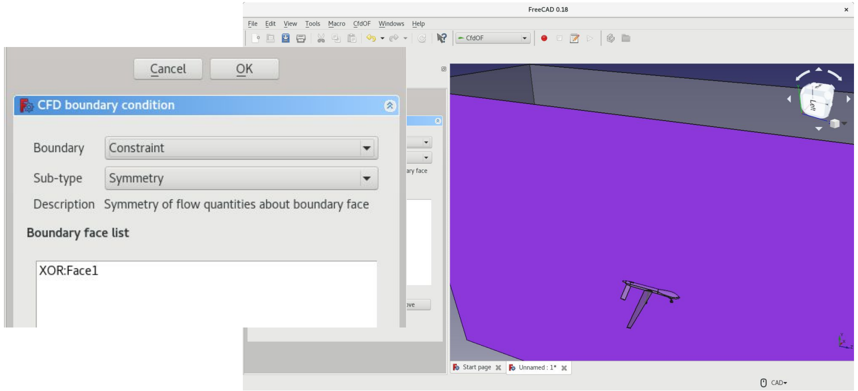
- To achieve a more gradual refinement from the far field to the surface of the body, we introduce an additional volume refinement
- Select the ‘Compound001_Mesh’ object and click the ‘Mesh region’ button as before
- Select ‘Internal volume’ and enter the parameters as shown
  - cfMesh currently only supports spherical and rectangular refinement zones
- Return to the mesh object and click ‘Mesh’.
- Click ‘Paraview’ to view the result.

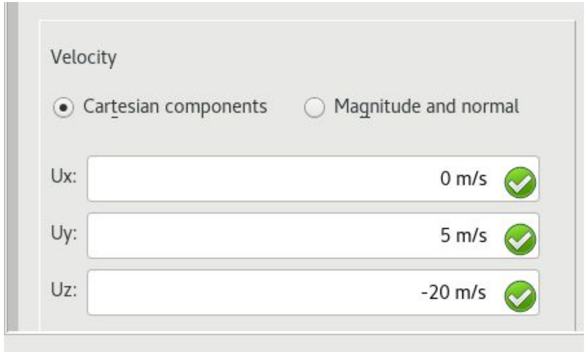


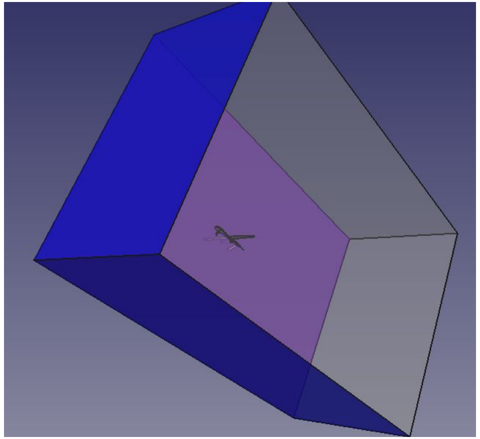
CFD analysis

Add boundary conditions

- Create a CFD boundary condition of 'Constraint' type 'Symmetry' for the central cutting face
- Create a 'Uniform velocity' inflow boundary condition with the parameters shown below and add the lower and front faces.
  - Flight at 74 km/h and 14° angle of attack

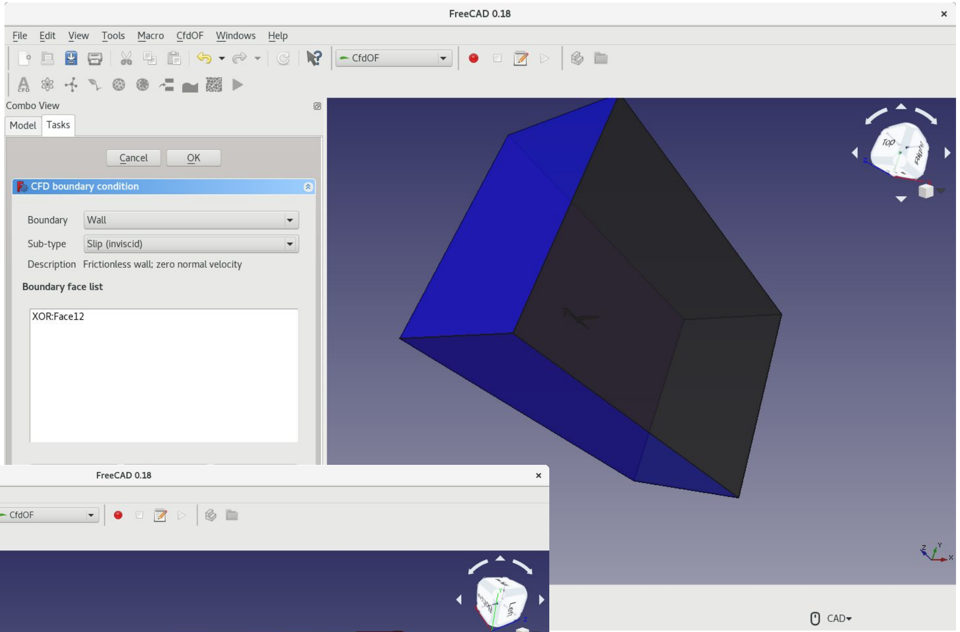


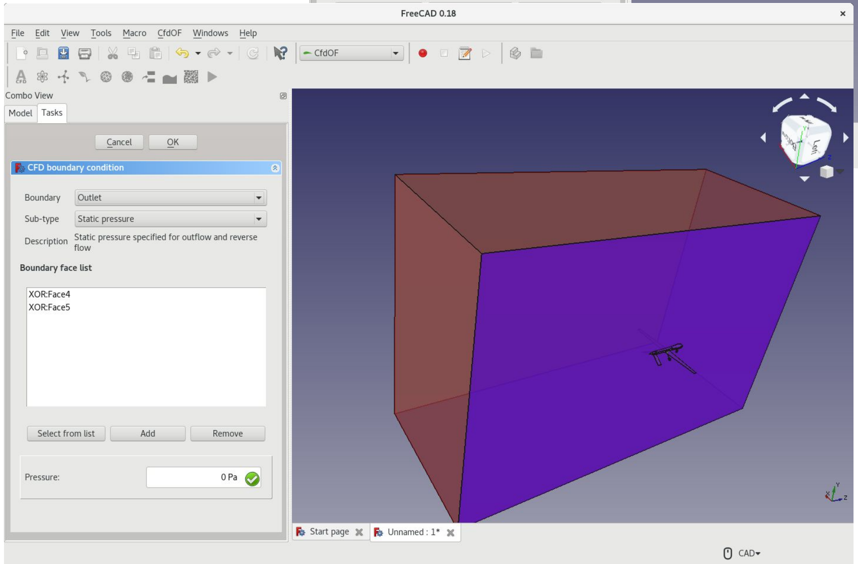




Add boundary conditions

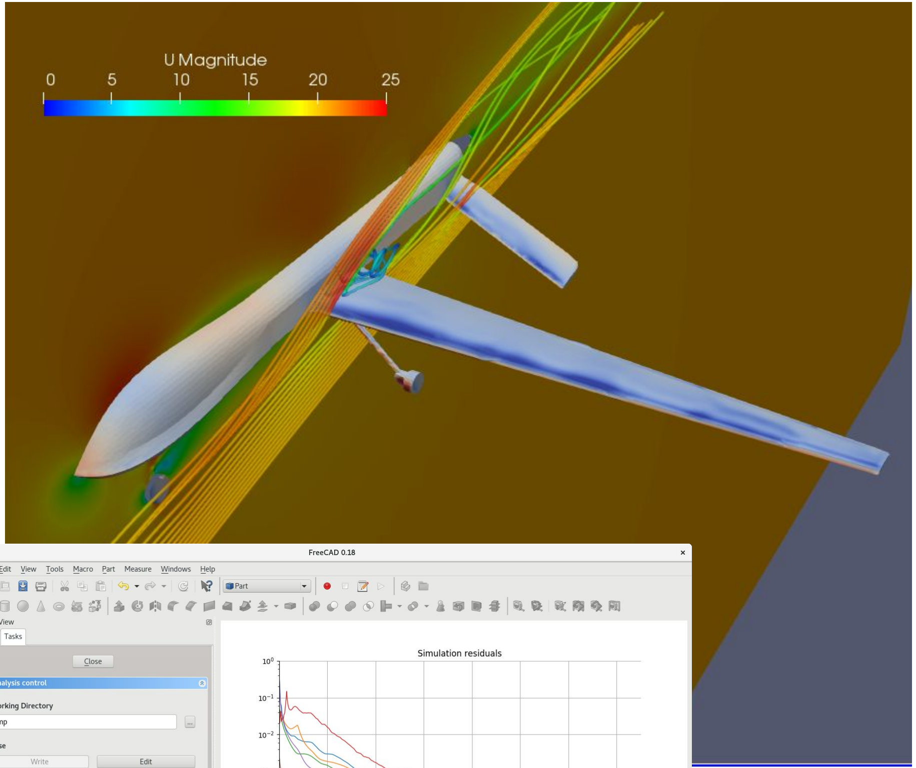
- Create a slip wall boundary condition for the outer face
- Create a 'Static pressure' outflow boundary and add the upper and back faces.
- The remaining settings include:
    - Fluid: air
    - Initialise with potential flow

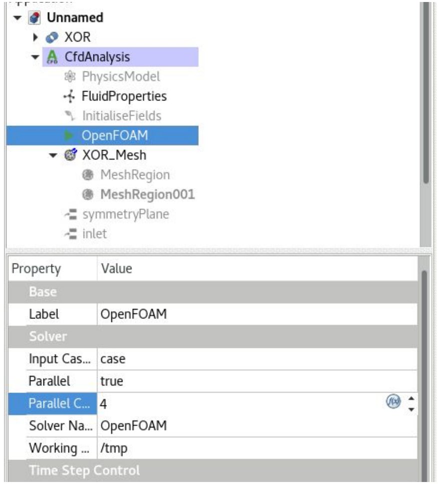


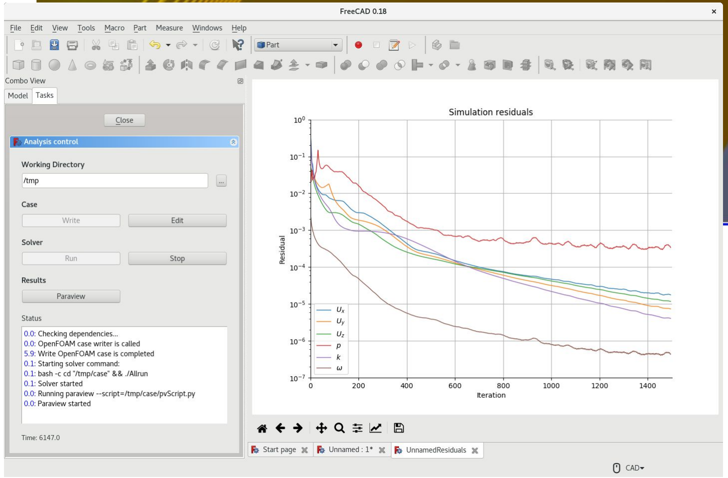


Run analysis

- For the ‘CfdSolver’ object, set parallel processing to true and the desired number of parallel cores
- Double click on the ‘CfdSolver’ object and click Write, then Run.



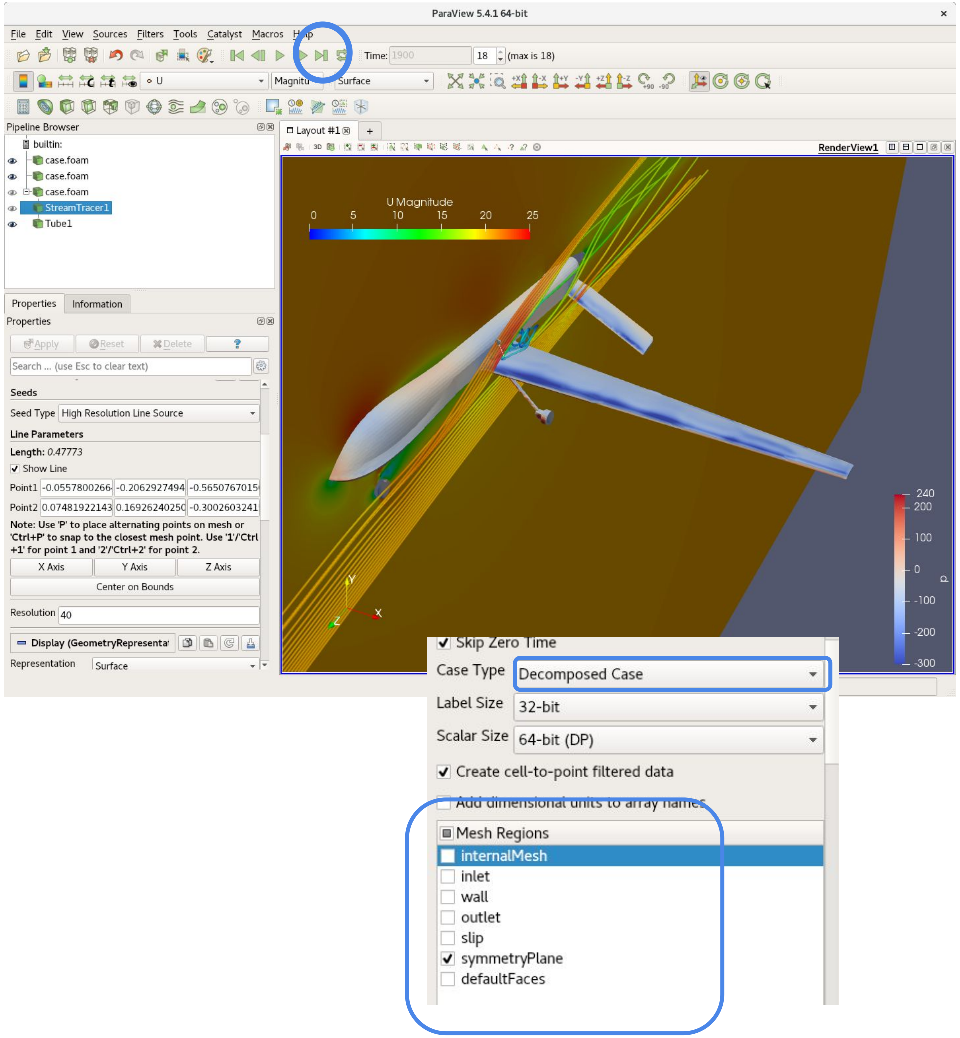




Post-processing

Visualisation

- Click ‘Paraview’, go to last time step
- Re-load ‘pv.foam’, select different patches to visualise if desired
- Add stream trace to object with internalMesh selected
- **Warning:** For incompressible, single phase solvers, OpenFOAM writes ‘Kinematic pressure’ = *p/ρ*



Integrated forces output

- While the analysis is running, click ‘Edit’ to open the case directory and edit the system/controlDict file.
- Paste the contents of the supplied ‘forces’ file at the end.
- Integrated forces and moments (about ‘CofR’) are written to the file:

  `postProcessing/forces_all/0/forces.dat`
- Use to determine lift, drag, etc
- Re-run analysis with and without camera gimbal, gear retracted, etc, and find effect on forces

```
runTimeModifiable true;

libs
(
    // Needed for availability of porous baffle boundary in
potentialFoam
    "libturbulenceModels.so"
);

functions
{
    forces_all
    {
        type            forces;
        libs            ( "libforces.so" );
        patches
        (
            wall
        );
        rho             rhoInf;
        rhoInf          1.2;
        log             off;
        writeControl    timeStep;
        writeInterval   1;
        CofR            ( 0 0 0 );
    }
}

// ****************************************************************
//
```

The End